



Gunthard Kraus, DG8GB

# An Interesting Program

## MSTRIP40

---

### 1.

#### Brief description

---

**Name:** MSTRIP40.ZIP

**Area of application:**

EM simulator using the moment method for the precise analysis of microstrip structures (main applications: calculation of patch antennas, slot antennas, couplers, etc.).

**Its the tops:**

The simulation automatically captures all physical effects and determines all important characteristics (e. g. in antennas the gain, the 3D radiated field pattern, any 2D diagram desired, the S Parameters, the cross polarisation etc.). Even slot antennas and structures with slot couplings can be correctly calculated, thanks to a refined trick.

**But theres a problem:**

In the present demo version, the simulation is restricted to double sided coated printed circuit boards (structures with an insulator plane and 2 metal planes). The full version creates multilayers with three insulator planes, but is no longer available free.

**File type and file size:**

Bundled program with download size of 2.6MB. Following installation, this gives a Windows program of about 8MB.

No problems were detected during operation using WIN 95/ 98/ME and a fast computer.

**Bugs or serious computing errors:**

In principle: No, but as soon as an entry was forgotten or the wrong entry was made anywhere, although there was generally no message concerning it, the program simply crashed with a runtime error message.

**Suggested improvement:**

Currently, printing is very poor DOS quality and is in sharp contrast to the lovely coloured graphics on the Windows screen. (Tip: you can use, for example, Paintshop Pro or some similar graphics program to help and to print the screen display directly).

**User friendliness:**

Easy to understand and familiarise yourself with, though some knowledge of basic principles is assumed with regard to the characteristics and limitations of modern EM simulators, and you need a lot of experience with such projects if the results are to be correct.



### **Aids:**

Online help in the form of the English language manual (40 pages), which can also be printed out separately. Moreover, enough specimen layouts are supplied (Demo1 to Demo21), all of which you must print out, debug and analyse precisely before starting your own designs. Also the programmer (Prof. Splitt of Kiel College of Higher Education) has placed all his scientific articles and presentations on associated subjects on the homepage ready to be downloaded. Demanding, but highly recommended!

### **Procurement options:**

Download from

<http://intek.e-technik.fhkiel.de/Splitt.htm>

---

## **2.**

### **Brief overall view**

---

All important characteristics and/or advantages can be found listed immediately in the first few lines of the manual. The extract reads as follows:

".....the program is intended for the analysis and design of any type of multi-layer microstrip structures or microstrip antennas. This involves rigorous work on the solving of the necessary integral equations using the Galerkin method (moments method), and the correct Greens functions are used. All essential effects, such as radiation or surface wave generation in the substrate, together with dielectric losses, are therefore taken into account in the simulation...."

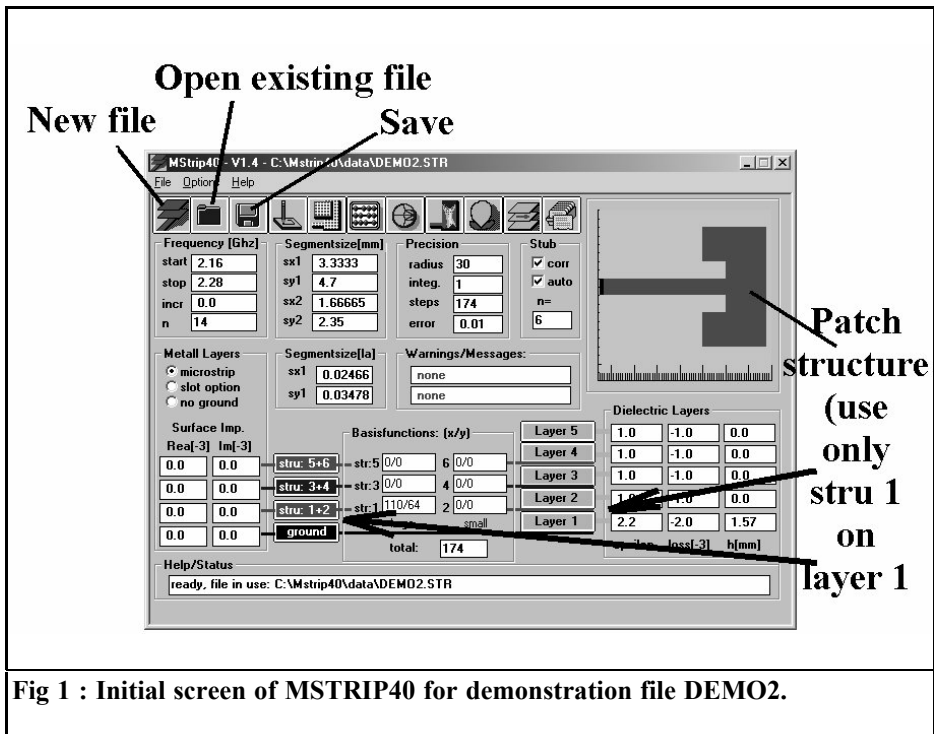
The user first has to divide the structure to be investigated into a lot of small but equal "cells" using known methods (the cells are called "segments" or "basic functions" here). Then a port is connected up and thus the circuit is activated, for example using a pulse func-

tion. The exact current density is determined in each element and then the reciprocal magnetic and electrical influences of the cells are calculated (here the assumed maximum "range" of the influence is 20 cells!). Finally the readings are totalled up to obtain the precise current distribution in the system, which leads to the determining of the electrical data such as radiation characteristics, input resistance etc.

But lets just leave aside the Galerkin method and the Greens functions used here. They can no longer be dispensed with in modern circuit design and nowadays they are used by many simulators and authors. The most important element in the foreword and in the method is actually that you can use it, for example, to obtain automatically all the radiation characteristics and electrical data for an antenna. This is also the case with the professional SONNET program, which we have already reviewed several times, but there these options were unfortunately blocked in the free "Lite" version. Thus the area of application of MSTRIP40 slowly becomes clear:

Everywhere where all radiation characteristics of an antenna are of interest, or where no substitution model for simulations is available in PUFF (e.g. slot antennas or patch antennas with slot coupling) or where we come up against the limitations of the SONNET Lite version, we reach for MSTRIP40. From my experience, however, the following things must be taken into account:

- The programmer recommends cell dimensions of between 5% and 20% of the wavelength. In practise they should not exceed 10%, as above that the errors increase. Most of the demo examples supplied have been simulated using cell dimensions of between 2% and 4%, and this clearly gives the best approximation to reality. You should not go down below 1%, as the calculation will take a very long time and the accuracy tends to



**Fig 1 : Initial screen of MSTRIP40 for demonstration file DEMO2.**

become progressively worse, even the program warns you about this. Moreover, if at all possible, a computer with a clock speed exceeding 300MHz should be utilised waiting around for results is just no fun at all. Another tip from experience: the height/width ratio of the two cell edges must lie roughly between 1:1 and 1:2 to retain the optimal precision of the simulation.

- Please do not make the sweep range too wide for the simulation. In my experience, the results can be trusted if the bandwidth is less than 30% of the mean frequency. This is connected with the fact that the “coupling of the activator signal into the structure” essentially generates a source of interference, which is compensated for by an automatically stub tuned to the mean frequency. Now we all know that matching using stubs is unfortunately a narrow band thing!
- For the same reason (interference due to coupling), you should never connect the input port directly to a structure, but connect it through a relatively long transmission line (e.g. with  $Z = 50 \Omega$ ).
- A maximum of two different cell sizes can be used. “Structure 1” simulates examples of  $S_x$  and  $S_y$  on Layer 1 accurately, using the segment dimensions entered. This involves a great deal of hit and miss work with your pocket calculator until you have established values which are appropriate for both the microstrip structure and for the feed at the same time, without any serious discrepancies. And naturally you need a good stripline calculator (my tip: TRL85 from ANSOFT), since Mstrip40 can only analyse and operate with standards set by the developer. “Structure 2” also operates on Layer 1, but displays exactly half the dimensions of structure 1 and



must have it's own drawing in the circuit editor. It is then made congruent by entering a substitute with structure 1. (See corresponding demo examples).

### 3.

## Procedure using program MSTRIP40

### 3.1. Installation

Following download the programs and data must be unzipped on the hard disc in the usual way. It should be unzipped to the directory C:\Mstrip40. In this new directory, we find all the necessary exe files, together with two important folders:

- “mstrip40\data” contains all the simulation examples supplied.
- “mstrip40\ manual” has the operating instructions and the online help.

Before we start the program, we add an attractive icon to the Windows screen (You already know how: click on the right hand mouse button in an empty spot

on the screen, then “New link”, “Scan”, select “Mstrip40.exe”, open.....etc.). But we still can't start the program. There is something very important to do first.

The newly unzipped “mstrip40.ini” file must be copied into the C:\WINDOWS directory straight away. As a precaution, you should then open it immediately with a text editor and check and/or correct the settings. Table 1 shows the correct and comprehensive path settings for storage on hard disk C:\ in directory “mstrip40” with an automatic start up of example “DEMO2”. The path for the personal HTML editor in the fifth line may need altering. Please check everything!

### 3.2. Demonstration of characteristics and operation using example DEMO2

Now we start up the program and we should see Fig. 1. Unfortunately the black and white text does not reflect the blaze of colour which welcomes you. In Fig. 1 it is immediately clear that we can not only start up a new project or store a change. Using the centre button, we can actually load the various examples directly, a click brings up a long list of structure files with the suffix “str”. We make the selection by clicking on the

```
[Mstrip40]
PathName=C:\Mstrip40\data\DEMO2,STR
FileName=DEMO2.STR
EditorName=notepad.exe
HtmlName=C:\Programme\Internet Explorer\EXPLORE.EXE C:\mstrip40\manual\manual.htm
ProgramPath=C:\MSTRIP40\

[Ms_smith]
PathNameC3d=C:\Mstrip40\data\DEMO2.slv
FileNameC3d=Mstrip40\data\DEMO2.slv

[Current3D]
PathNameC3d=C:\Mstrip40\data\DEMO2.slv
FileNameC3d=Mstrip40\data\DEMO2.slv

[Pattern3D]
PathNameC3d=C:\Mstrip40\data\DEMO2.slv
FileNameC3d=Mstrip40\data\DEMO2.slv
```

**Table 1 : Contents of the Mstrip40 control file Mstrip40.ini.**



corresponding file. The example is automatically loaded and the one previously in use is overwritten.

Please just call up various examples one after another and marvel at their structure, but return to "DEMO2.str" at the end!

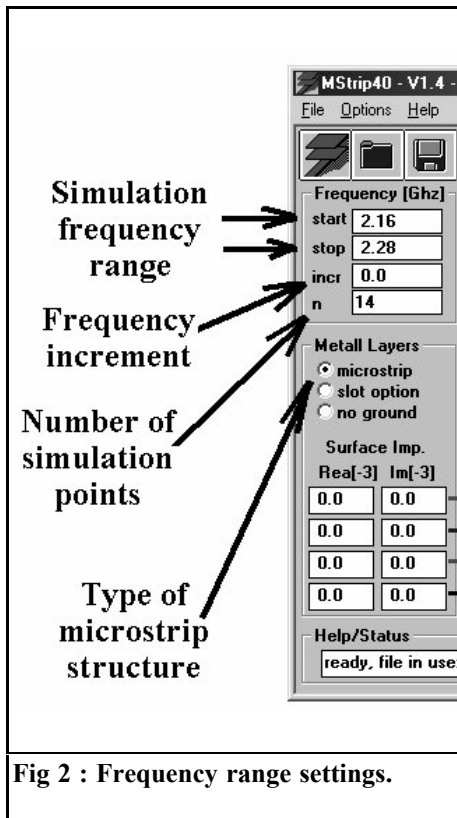
Lets start by examining the construction of DEMO2, which has to be analysed by the program. We are dealing with a patch antenna for 2.2GHz, in which the feeder goes directly to the optimal matching point and so the line structure of the antenna is used for transformation. This method is known from [1]. The main difference from the design at that time is the notch in the patch to connect the microstrip directly with the feed point. This is a simulation problem that PUFF cannot handle because it has no suitable model.

If you look more closely at Fig. 1, you can quickly make out the details of the structure: a printed circuit board material is used with a dielectric constant of 2.2, a loss factor of 0.002 and a thickness of 1.57 mm. The board underside is provided with a complete earthing surface and only large "basic functions" (elements or segments or cells) are used.

In Fig. 2 we see the settings of the frequency range to be swept. The user can pre-set, not only the start and stop frequencies, but also the number of simulation points. The frequency increment between two points is then automatically adjusted (provided the increment selected is not too fine, then only "0.0" is displayed). In addition, the evaluation option can be chosen for the special features of the structure to be examined (Microstrip / Slot / No ground).

Fig. 3 shows the entry fields for the segment sizes. Its worth remembering the following details:

- The edge lengths  $S_x$  and  $S_y$  may not exceed 20% of the wavelength, otherwise there are considerable errors. Dimensions below 5% of the wave-



**Fig 2 : Frequency range settings.**

length increase the accuracy, but increase the computing time and should therefore be used with care (e. g. when the computer used has a clock speed exceeding 1GHz). Optimal dimensions are those which are approximately 2 to 3% of the wavelength.

- For better monitoring, therefore, the values of  $S_{x1}$  and  $S_{y1}$  (the "coarse" cells) are also based on the wavelength and are displayed separated in two windows.
- The  $S_{x2}$  and  $S_{y2}$  dimensions of the "fine" segments are always exactly 50% of  $S_{x1}$  and  $S_{y1}$ . This ratio can not be altered.

In Fig. 4 the settings for the simulation accuracy are shown on the left of the circuit. The definitions are as follows:

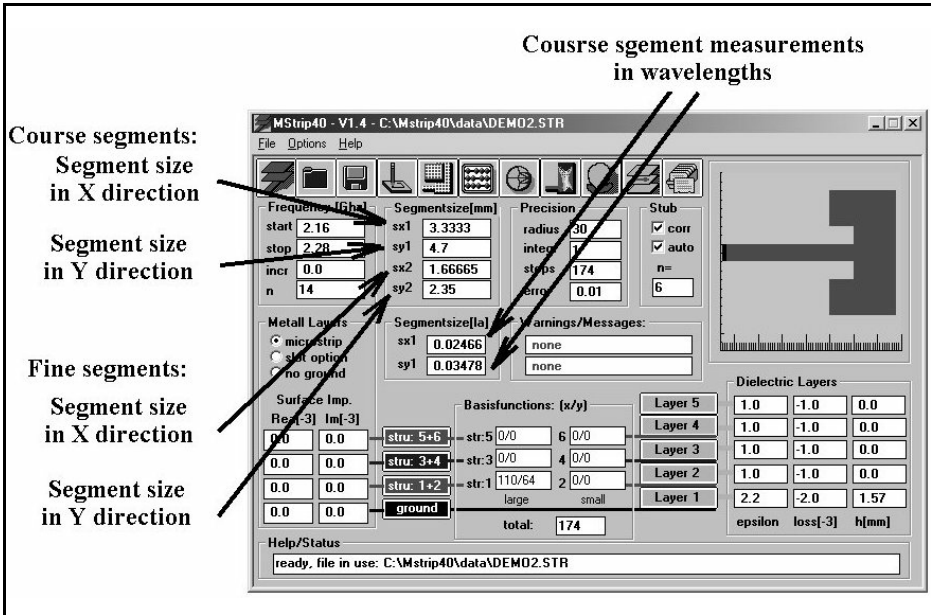


Fig 3 : Entering segment size information.

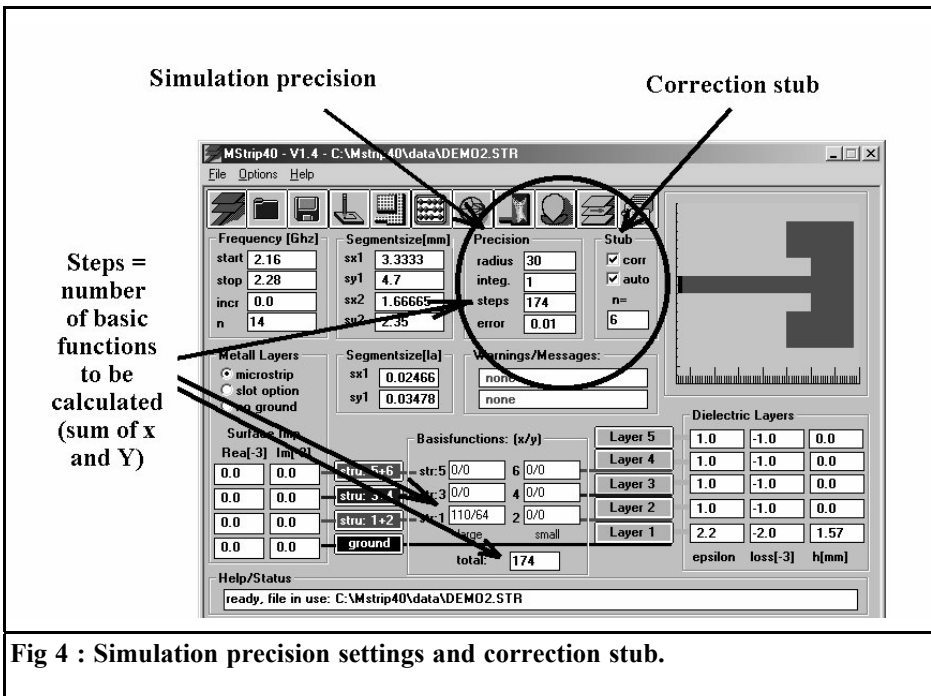
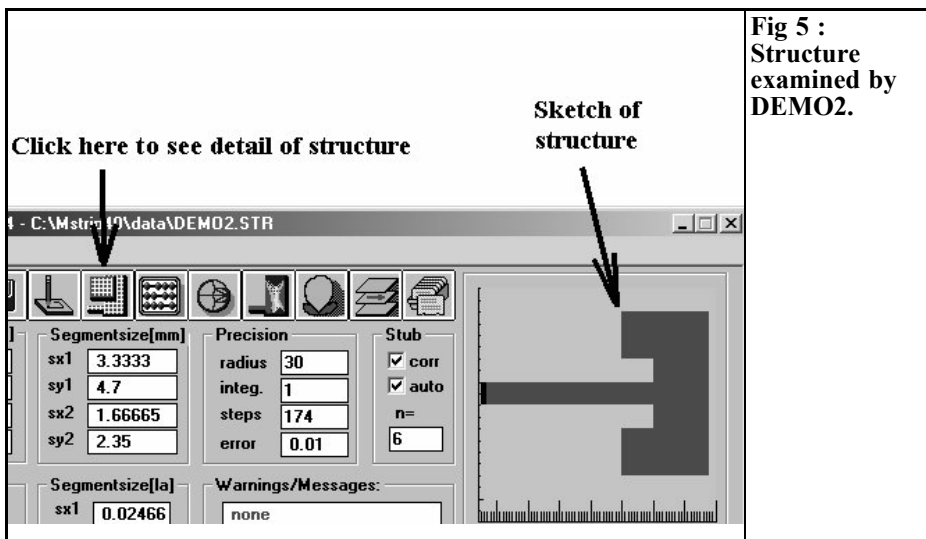
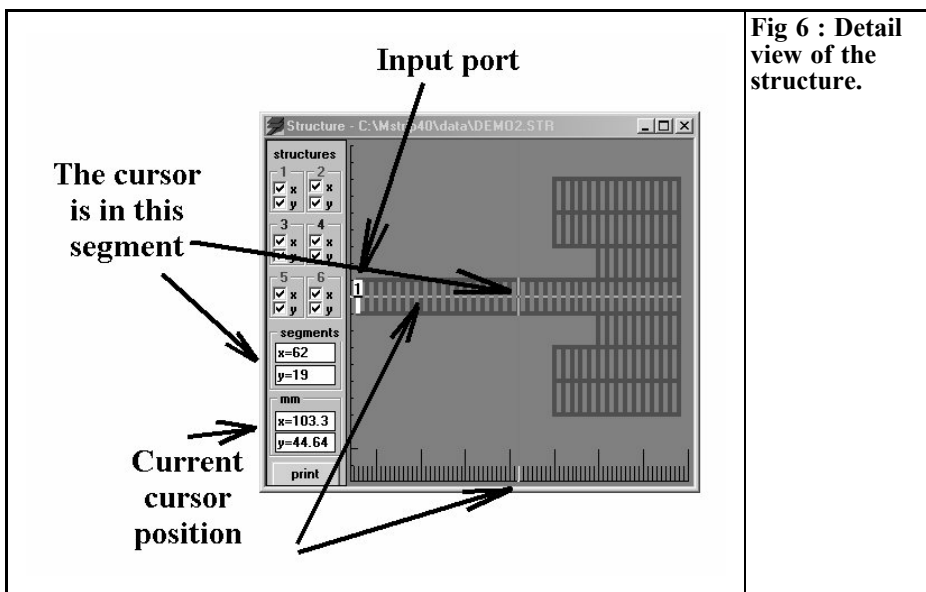


Fig 4 : Simulation precision settings and correction stub.



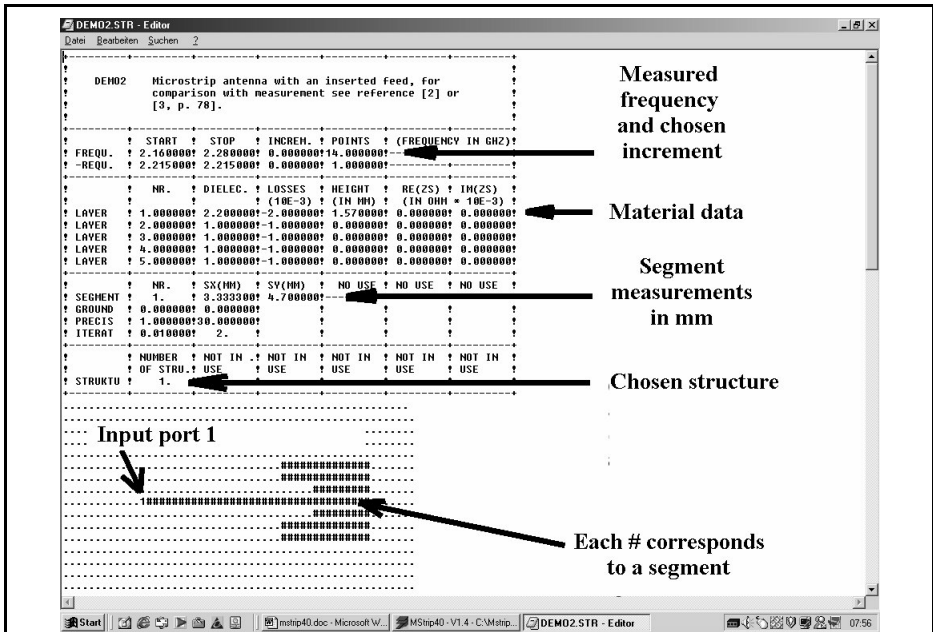
**Fig 5 :  
Structure  
examined by  
DEMO2.**

- The “Precision Radius” determines (in the form of the number of segments) the “range of reciprocal influencing and coupling” in a cell. On this point, the manual supplies the information that sufficiently precise results can be expected if this radius covers two-thirds of the structure to be investigated.
- The “Integration Precision” factor is nominally 1 and lays down the numerical computing accuracy in solving the reaction integrals. Only for extremely small or extremely large basic functions or for very thick layers can it be increased up to 5 (once again at the expense of computing time).



**Fig 6 : Detail  
view of the  
structure.**





**Fig 7 : ASCII text file containing all of the simulation information.**

- “Steps” indicates the number of basic functions to be processed. In each case, the number includes all the individual segment edges present in the X direction (here: 110) and the number of segment edges in the Y direction (here: 64).
- “Error” gives the permissible residual error for the calculation of such an integral (Default: 1%). If it is under-shot, the program moves on to the next basic function.

On the right hand side of the circuit there is a stub option, this compensates for input impedance errors caused by the input signal. As soon as “Automatic” is activated, the program automatically corrects this problem by connecting an open stub (not visible). But please remember that such compensation is narrow band, and heed the warnings in the manual (Page 11):

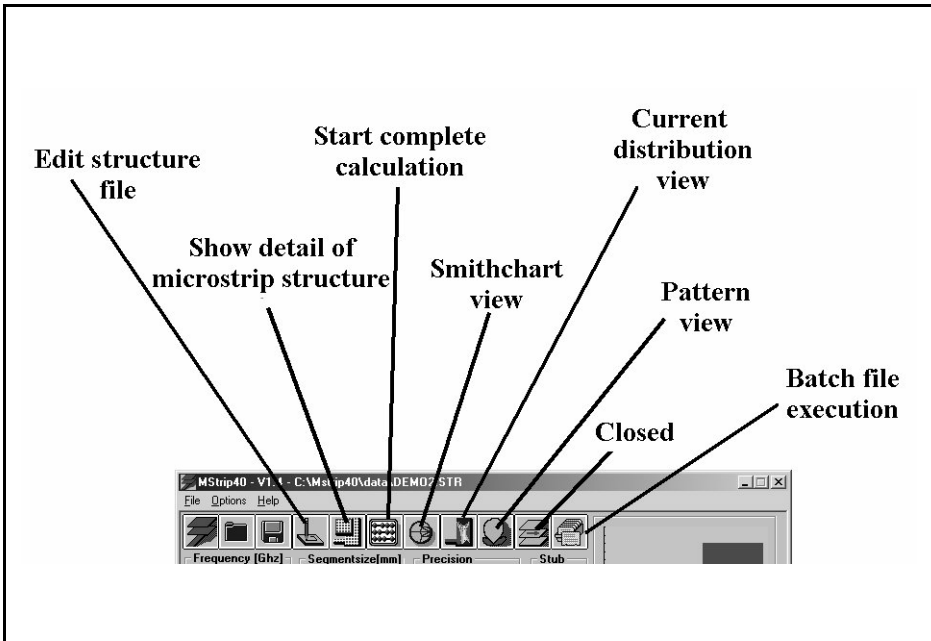
- “Dont select frequency ranges which are too big!”

- Leave sufficient distance between the coupling port and the structure (using a suitable feed line)!

Now can we pay closer attention to example DEMO2.

The antenna structure which can be seen on the right hand side is used only for rough information, the precise dimensions and a precise breakdown of the cells used can be seen by clicking on the button shown in Fig 5. This opens a window, which can be zoomed, in this way the precise structure can be examined, see Fig. 6. Take a really close look and you will see something nice: the program automatically breaks each cell up again into four equal sections, in order to increase the accuracy of the simulation this can be seen clearly as the cursor is moved around. One last tip: the small white box with the number “1” identifies “Port 1”, which is used to activate the structure using the input signal.





**Fig 8 : The function of remaining buttons.**

### **Very important:**

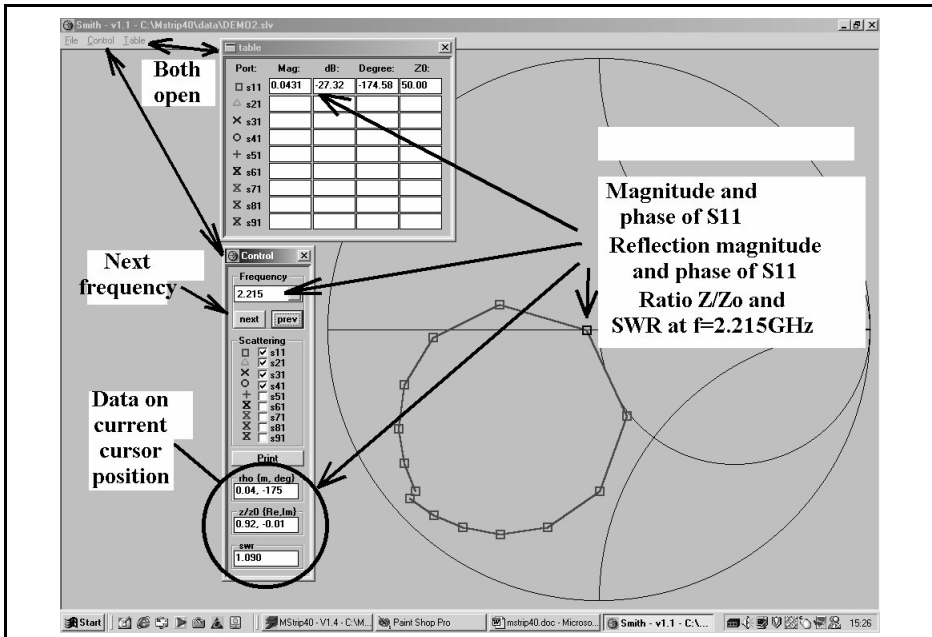
This display is only there to provide precise information to the user and can not be altered. All the details, together with the type of structure or data entry, can be seen only if we press the button with the pencil (it is directly to the left of the button just used). In Fig. 7 we can see that, amazingly, everything is done with only a simple ASCII file as used by any text processing program! Its easy to see the values transferred from the screen input here for the frequency range being examined, the printed circuit board data, the segment dimensions and the structure 1 selected for this project (largest segments on layer 1). The selected entry form for the microstrip structure is especially amusing, it is embedded in a field of dots (these form, so to speak, the printed circuit board with the lower continuous earthing surface, i.e. the “ground” and can be imagined as going on for ever). Each hash sign entered

represents a segment and the “1” at the beginning of the feeder is naturally port 1 for activating the structure.

### **A tip:**

Those wishing to experiment further should make life easier for themselves and store the demo example under another name. Then open this structure file and experiment by altering the values entered or the “drawing”. In this way, you can save yourself the trouble of starting a new file and keying in all the variables required that would take up a lot of time!

If we now look at Fig. 8, we find the functions of all the remaining buttons explained. Anyone with a lot of time to spare can re-start the entire calculation using the next button in the row (symbol: little computer) on relatively old computers, this is an opportunity for an extended coffee break! It is interesting to print out the Smith chart of the results (Fig. 9). As



**Fig 9 : The Smithchart view, once displayed open the "Control" and "Table" menus.**

soon as you zoom the graphics to fill the screen, you have to open the two pull-down menus "Control" and "Table". "Table" shows the simulated S-parameters in table form for the frequency just set, whilst additional useful information is supplied in "Control". The buttons "next" and "previous" can be used, for example, to alter the frequency, with the active measurement point being brought out in black on the Smith chart.

There is something more to say about the cursor display, marked with a circle (reflection factor with level and phase, SWR and  $Z/Z_0$  split into real and imaginary sections). The data displayed within the circle is only valid for the frequency actually selected if the cursor has not left the "Control" menu area. As soon as you roll over the edge into the Smith chart, the values associated with the current cursor position on the screen (based on the centre point of the Smith diagram!) are displayed. Definitely something you have to get used to, because rolling back

to "Control" does not change the display back again. You actually have to click on "prev" or "next" to switch the display back to a value from a table!

Now click on the button for current distribution. Fig. 10 cannot not show the beautiful colours on the screen, but the basic graphics principle is easy to see, as well as the additional options. Not only can you call up the Smith chart graphics onto the screen as an aid to selecting the desired frequency, but you can also call up a 3D radiated field pattern (Fig. 11). First, you select the structure to be examined (here structures 1+2, i.e. large and small segments on layer 1). You can choose between black and white or colour representation, and rotate the image using the four arrow keys. Looking at the display you will immediately see the two enormous peaks in the current density. The current from the feed connection flows in the direction of the radiating patch edges and round the corners of the notch. The display enables you to see

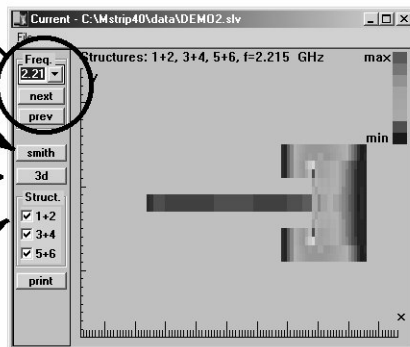


Frequency  
selection

Smithchart  
view

3D view

Select  
structure  
required

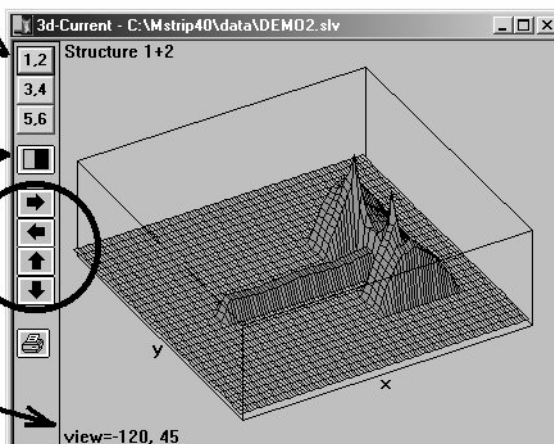


**Fig 10 : The  
current  
distribution at  
the selected  
frequency.**

Select  
structure  
required

Black and  
white or  
colour

Adjust  
viewing  
angle



**Fig 11 : 3D view of the current distribution.**

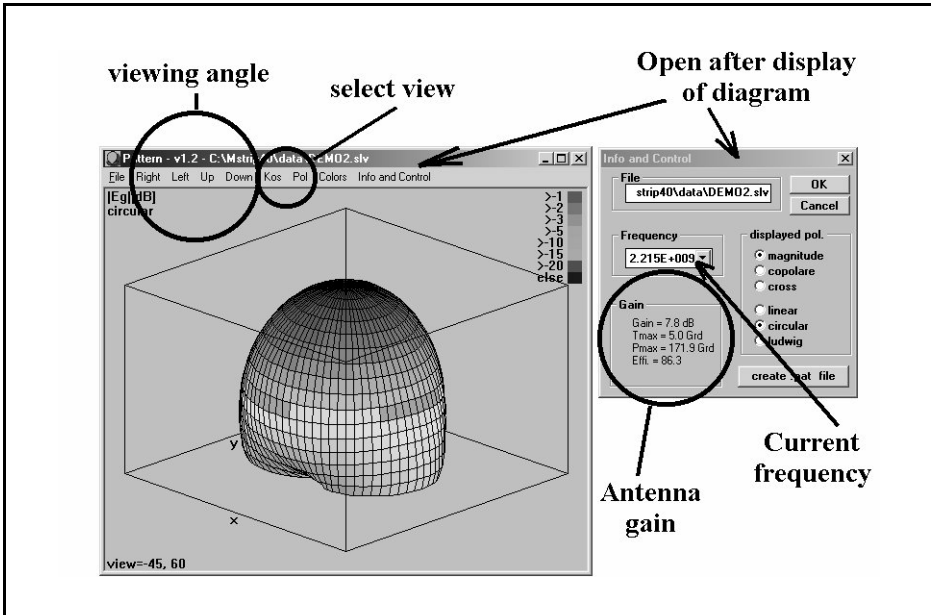


Fig 12 : Radiation pattern view with information on antenna gain.

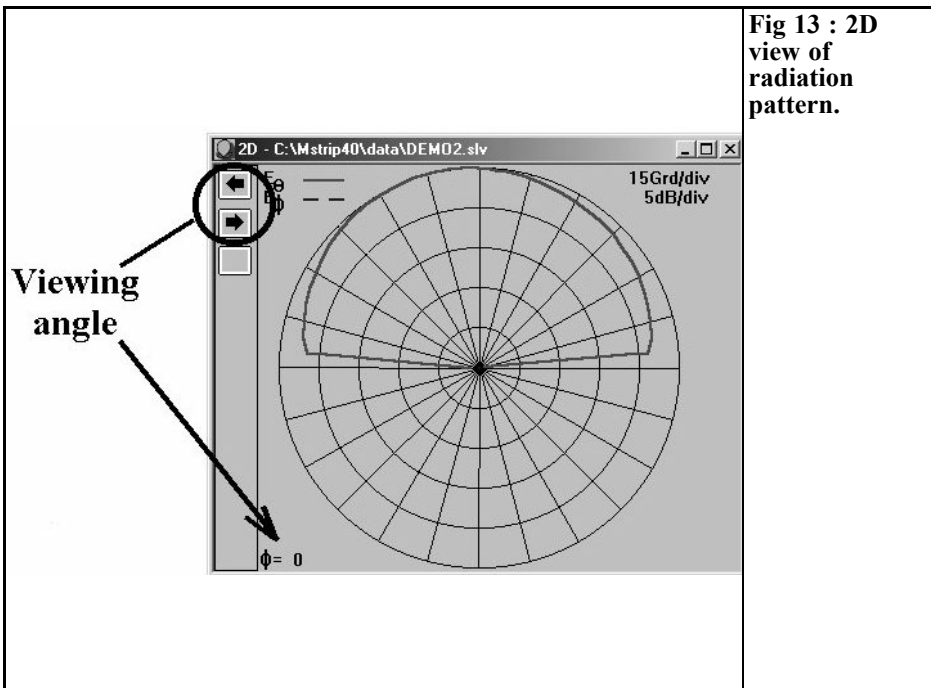
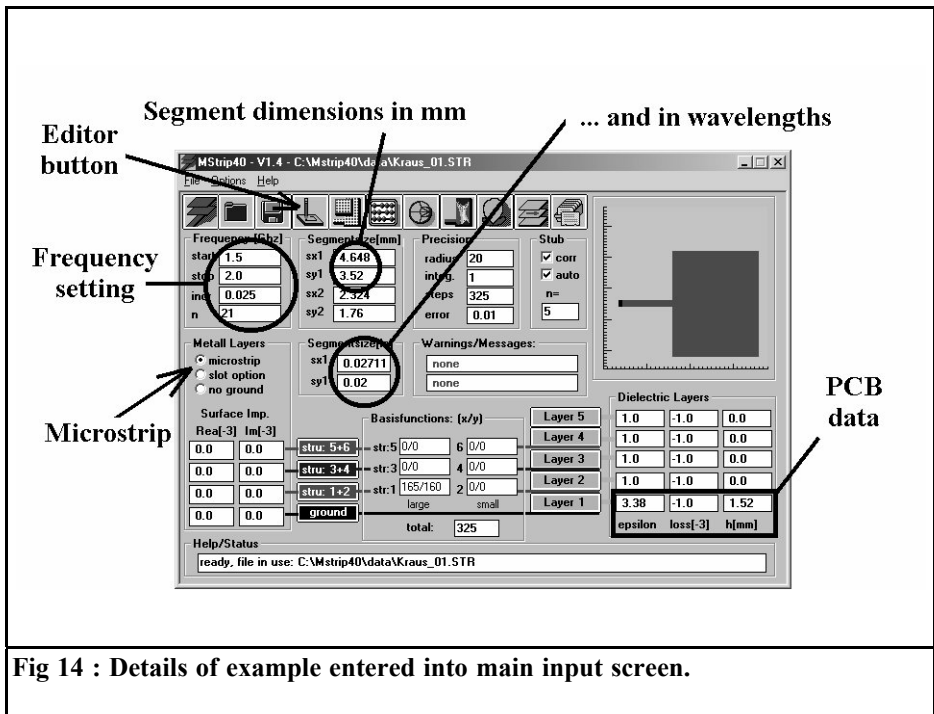


Fig 13 : 2D view of radiation pattern.



**Fig 14 : Details of example entered into main input screen.**

exactly what happens.

The next button brings us to the radiation pattern diagrams. Once the menu is opened, call up “Infos and Controls” and immediately read off the prediction for the antenna gain (Fig. 12) together with the details of the 3D radiated field pattern. The four arrow keys can again be used for rotation i.e. the angle of observation can be changed. But you should just play about with the various simulation options (right up to cross polarisation). To be honest, for individual menu items you really need to look through the manual or look in the appropriate literature in order to grasp the deeper meaning.

### Incidentally:

If you miss your beloved 2D directional diagram, click the option “2D Pattern” under “Kos”. Fig. 13 will then make you very happy, since here too the angle of observation can be varied using the ar-

row keys.

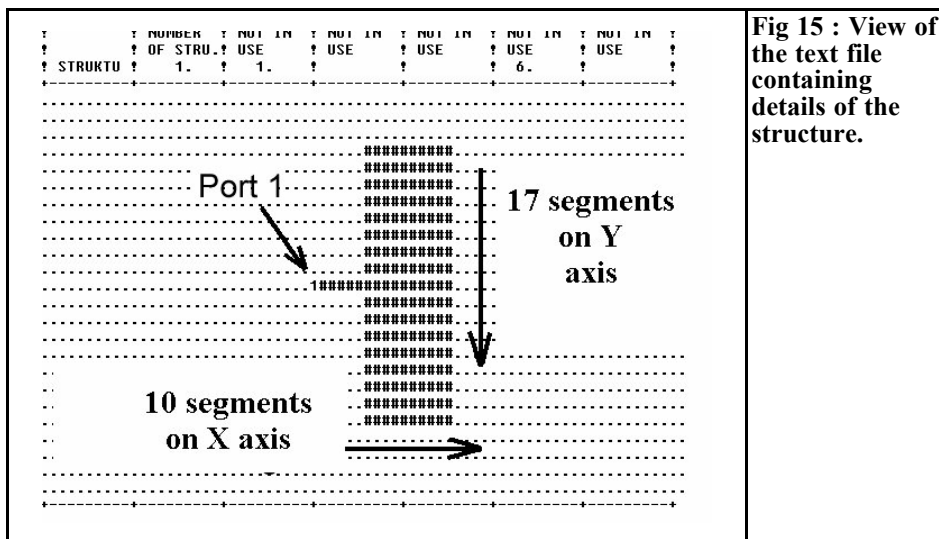
### 3.3. My own example:

Determining the data for a patch antenna for METEOSAT

As a stimulation for your own work, the specimen antenna from the article in VHF Communications [1] is to be simulated here using Mstrip40, to check how far we are from the actual measurement taken at the time. The material data are as follows:

- Printed circuit board material: Rogers R04003
- Relative permittivity: 3.38
- Board thickness: 1.52mm
- Dielectric loss factor  $\tan\delta$ : 0.001 at 1,600MHz

The patch created and investigated had a width of 58.67mm and a length of 46.48mm, this length giving precisely the frequency of resonance of 1,691MHz (see simulation in Fig. 31 on Page 76 in



**Fig 15 : View of the text file containing details of the structure.**

Part 2 of the article referred to). For a feeder with  $Z=50\ \Omega$ , we need a microstrip line with a width of 3.52 mm.

### 1st step:

Open the specimen "DEMO1.str" and save it using "SAVE AS", giving it a new personal name.

### 2nd step:

Enter the board data correctly, split the frequency range from 1.5 to 2GHz into 21 segments and adopt the precise width of the  $50\Omega$  feeder (i.e. 3.52 mm) for the segment dimension,  $S_y$ . Divide the radiator length of 46.48 mm into 10 equal sections, thus giving a value of  $S_x = 4.648$  mm.

Very important: you must save everything immediately afterwards by "pressing the key with the diskette".

Fig. 14 shows the screen with these entries. Both the step width for the frequency steps and the values of  $S_x$  and  $S_y$  based on the wavelength are entered automatically. It can also be seen that the requirements listed at the beginning of this article (dimensions between 2% and 4% of the wavelength, and no side ratio exceeding 1:2) are correctly fulfilled.

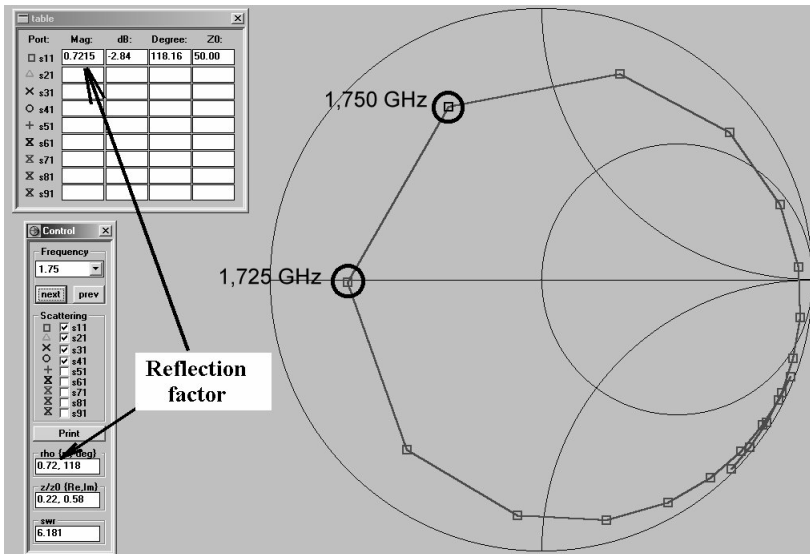
### 3rd step:

Now open the circuit editor with the specified button and look at the top half of all the simulation objectives.

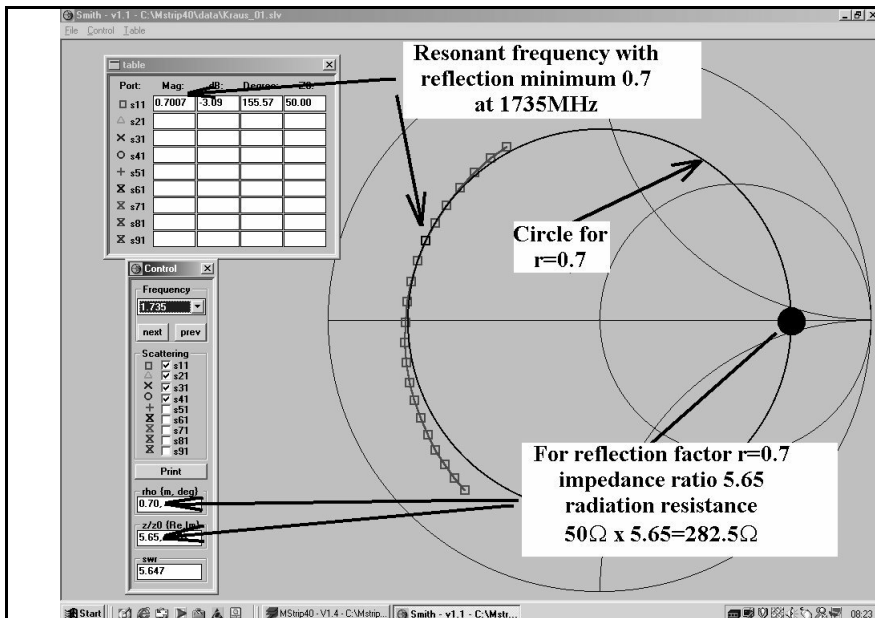
The antenna is in the bottom half of the page change the existing structure there with the keyboard. In the horizontal direction, enter 10 hash signs in each line (10 x 4.648 mm = 46.48 mm). There is an inaccuracy in the vertical direction,  $S_y = 3.52$  mm and for a patch width of 58.67 mm we would actually need 16.66 segments. So we use 17 by filling 17 lines with a hash signs. Finally, please don't forget the feeder (in line 9) and the "1" for the input port. Fig. 15 shows what the result should look like.

### 4th step:

First carefully save everything and then close the editor. Press the "key with the computer" to start the simulation and, following an appropriate compulsory coffee break, call up the Smith chart graphics for the S-parameters. After calling up "Control" and "Table" as well (as previously described), you should have a graphic similar to Fig. 16. The input reflection loop, which is typical of patch antennas, is very easy to recognise, and



**Fig 16 : The first simulation shows the resonant frequency of the antenna between 1725 and 1750MHz.**



**Fig 17 : The narrowband simulation show the resonant frequency of the antenna as 1735MHz and the radiation resistance as 282.5Ω.**





you have only to look for the minimum of  $|S_{11}|$  to find the natural resonance. According to the simulation, it lies somewhere between 1.725 and 1.75 GHz and unfortunately the selected scale division is too coarse there. Consequently, we use a new simulation with a narrower frequency range of 1.700 to 1.750 GHz. Please don't forget to save the new entries again before starting the simulation on the basis of my experience, it is perfectly reasonable to open the editor files again and check whether everything has been correctly transferred. The program frequently develops an unexpected life of its own and simply rejects such commands, but does not report this on the screen. Now and then, though, the message flashes up that, if there are problems or crashes, the two files `"*.rea"` and `"*.slv"` should be deleted from the specimen directory with Explorer to make everything work correctly again.

### 5th step:

In Fig. 17 we can see not only the result of this narrowband simulation, but also the next action we have to take. We now call up the individual frequencies one after another and look for the smallest value of  $|S_{11}|$ . This can be found at the frequency  $f = 1,735\text{MHz}$ , where there is a minimum value of  $r=0.7$ . This can be used immediately, just roll the cursor along the horizontal axis of the Smith chart from its centre point, moving right until you reach the display `"rho = 0.07"`. The ratio of the real component to the system resistance is then automatically displayed in the `"Control"` field as 5.65. The patch antenna consequently displays a real input resistance of  $50\Omega \times 5.65 = 282.5\Omega$  at the feed point, since by this action we have simply `"turned back"` the  $50\Omega$  feed line right up to the radiating patch edge. If you take another look at the screen photo, Fig 28 on page 73 of VHF Communications 2/2002, it shows the network with its reflection factor of precisely 10% at the resonance point. Then do some calculations, you will

come up with an interesting result. The actual radiation resistance at the feed point of the antenna is actually 20 % higher here than it was assumed to be. If the start value selected for the simulation is  $200\Omega$ , in reality this would be  $1.2 \times 200\Omega = 240\Omega$  - so were not all that far away with the Mstrip40 simulation of approximately  $280\Omega$ . Thus the overall result is entirely usable, even if Mstrip40 also yields a self resonant frequency of 1,735MHz instead of 1,691MHz (which is just 3 % too high).

One more interesting thing: If we now repeat the simulation with a shorter or longer feeder, or with shorter or longer time measurements, then we are nevertheless still faced with a result which gives us a frequency of resonance of between 1,720 and 1,750MHz and an input resistance of between 250 and  $300\Omega$ .

### 6th step:

What would all this trouble be worth without the antenna gain? Correctly calculating the radiation characteristics is certainly one of the most important properties of Mstrip40. If you press the `"Pattern key"` in the main menu, and then open `"Info and Control"` and enter a frequency of 1,735MHz, you obtain the information `"Gain = 6.6 dB"`.

You can then also analyse the 2D and 3D directional diagrams in any way you wish, or check the current distribution on the patch at the precise resonance.

## 3.4. Summary

Mstrip40 only supplies really good analyses of the extremely diverse microstrip and antenna structures if the procedures discussed are adhered to. Solid knowledge regarding the subject matter is indispensable here, not just with regard to the objects to be simulated, their main behavioural characteristics and the results to be expected, but likewise with respect to the characteristics and limitations of the EM simulators. Otherwise,



the simulation result easily becomes a lottery! Before beginning any work of your own, it is indispensable to carry out a very thorough analysis of all the specimen programs and of the manual. This is the only way to get to the bottom of the mysteries of the simulation of normal and abnormal patch antennas, slit antennas, arrays, or any type of aperture-coupled structures, etc. Who would have previously suspected that, for example, the option “no ground” could have been used to simulate the main outlines of the behaviour of microwave dipole antennas in the open air? All in all, this requires a great deal of preparatory work, but it is rewarded with a considerably deeper understanding of the relationships involved and with usable preliminary plans for a practical project.

---

#### 4. Literature:

---

- [1] Gunthard Kraus: “Modern Patch Antenna Designs, Part 1 and Part 2”. VHF Communications 1/2001 and 2/2001
- [2] Mstrip40 manual
- [3] SONNET-LITE manual
- [4] APLAC manual